NASA TM-85550

NASA-TM-85550 19840018593

A Reproduced Copy
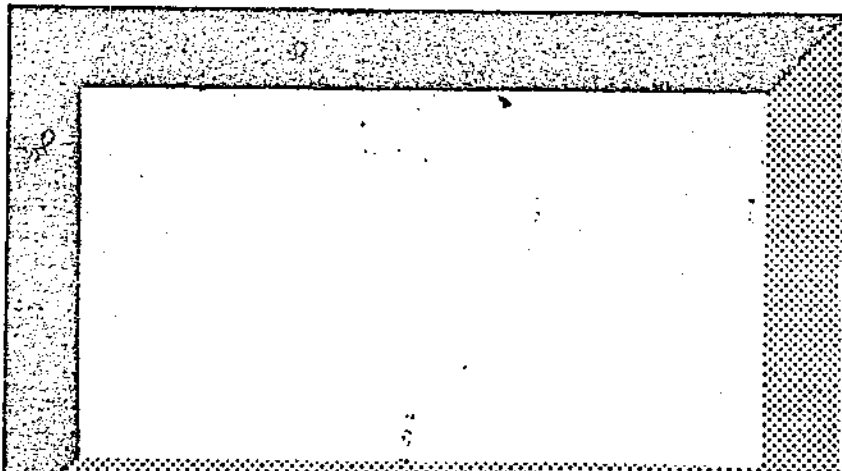
OF

NASA TM- 85550

Reproduced for NASA

by the

NASA Scientific and Technical Information Facility

FFNo 672 Aug 65

NAG-2-115

(NASA-TM-85550)   AERODYNAMIC DESIGN USING   N84-26661
NUMERICAL OPTIMIZATION (NASA)   34 p
HC A03/MF A01   CSCL 01A
                                                    Unclas
                                        G3/02   13645

DEPARTMENT OF
AERONAUTICS AND ASTRONAUTICS
MASSACHUSETTS INSTITUTE OF TECHNOLOGY
CAMBRIDGE, MASSACHUSETTS 02139

N84-26661

AERODYNAMIC DESIGN
USING NUMERICAL OPTIMIZATION

by
Earll M. Murman*
and
Gary T. Chapman**

CFDL-TR-83-2   February 1983

*Professor, Department of Aeronautics and Astronautics, MIT,
  Cambridge, MA 02139
**Senior Staff Scientist, NASA Ames Research Center,
  Moffett Field, CA 94035

Department of Aeronautics and Astronautics
Massachusetts Institute of Technology
Cambridge, Massachusetts 02139

The design of a flight vehicle is a very complex problem. Technical, economic, marketing and other challenges must be overcome to achieve success. It is not surprising that "design" means different things to different people. To the aerodynamicist it usually means "given a set of criteria, find the best configuration to achieve them." Even if we limit our consideration to "aerodynamic design," we are still faced with a complex problem. A team of people must use a combination of intuition, experience, testing, numerical and theoretical analysis to develop a successful design. The numerical part of this process has improved substantially in the past decade due to the development of computational methods which can treat realistic geometries and governing equations. A perspective of this development is given by Graves[1]. This paper discusses one computational approach to aerodynamic design, namely the procedure of using numerical optimization methods coupled with computational fluid dynamic (CFD) codes.

Perhaps we should first define computational fluid dynamics. CFD is the numerical solution of a discrete approximation to the partial differential equations describing the fluid flow past a body. The governing equations may be potential, Euler, Reynolds averaged Navier-Stokes or boundary layer, depending upon the important flow phenomena. Usually finite difference or finite volume methods are used, but other discrete approximations (finite element, spectral) may be adopted. For subsonic or supersonic inviscid flow the potential equation can also be solved by panel methods. For an aircraft configuration these computations take minutes on the biggest computers available. However, the entire solution of the flow field is obtained, including the aerodynamic forces.

Although the development of accurate, reliable, and economical CFD methods is far from complete, the ones which are available today can already treat many practical problems. Two-dimensional transonic airfoils are

routinely analyzed on the computer using methods pioneered by Bauer, Garabedian, Korn and Jameson[2]. Resulting airfoils are then tested in the wind tunnel to verify the design and to establish off-design performance. For 3-D configurations, panel methods are well developed and widely used for subsonic and supersonic analysis. The 3-D transonic methods are rapidly developing into reliable and accurate codes[3]. By way of illustration, F-1 shows calculations by Boppe and Aidala[4] for the shuttle launch configuration, F-2 presents calculations by Yu[5] for the effect of a body on the transonic wing pressure distribution for a transport configuration, and F-3 shows the shock pattern on a transonic compressor rotor calculated by Thompkins[6]. It can only be expected that in upcoming years the development of better algorithms and computers will produce improved methods for analyzing complex aerodynamic problems.

The problem of interest to us here is the utilization of emerging CFD capability for aerodynamic design. A number of approaches and techniques are being developed or used. The easiest and most commonly found one is a numerical "cut-and-try" procedure using a direct analysis code. As new vehicle geometries are conceived, the aerodynamic performance is computed. In other words, CFD codes are used instead of wind tunnel tests to arrive at an acceptable design. This procedure has been incorporated in the design of several new aircraft with substantial savings of wind tunnel testing hours. F-4 illustrates this for the Grumman Gulfstream III.

A second approach to the design problem is the "inverse method." As new aerodynamic understanding is developed, a pressure distribution is conceived and the geometry which corresponds to it is computed. Recent developments of these techniques are given in R-7-11. One difficulty with inverse

techniques is insuring that realistic geometries are obtained for the
specified pressure distribution. There are theoretical limitations for
two-dimensional flow on what constraints may be specified[12]. Generally,
the requirement of exactly matching the specified pressure distribution is
replaced by a relaxed requirement which yields a practical geometry and
some least squares best match to the specified pressure distribution.
Another uncertainty in inverse methods is knowing which pressure distribution
is really optimum. For example, should it contain a shock or be shockless?
Garabedian and McFadden[11] found that specifying a shockless pressure dis-
tribution on a swept wing did not lead to a shockless flow field solution.
"Hanging shocks" were found to exist above the body which weakened before
reaching the body. An optimization procedure was then used to modify the
surface pressure distribution to achieve nearly shockless flow.

There are a number of design techniques which are used to varying
degrees but apply to a limited class of problems due to the basic assumptions
inherent in the formulation. For example, hodograph methods[2,13] are extremely
useful for two-dimensional, shock-free shapes but they are not extendable to
more general problems. Chin and Rizzetta's[14] streamfunction method can solve
two-dimensional inverse problems with shocks, but extension to three-dimensional
problems is not clear. The clever artificial gas method of Sobieczky is
applicable to two- and three-dimensional shock-free shapes[15]. Other methods
exist which we have not mentioned.

Another approach to design is to seek optimum shapes which satisfy a
stated design objective. This approach was highly developed using calculus
of variations for problems which could be treated by classical theoretical
aerodynamics. The book by Miele[16] represents the extent to which this approach
has been developed. An "optimum body" is a shape which has the maximum or

minimum value of a desired objective function, subject to specified constraints. For example, elliptic wing loading yields the minimum induced drag for a given span, and a Sears-Haack axisymmetric body has the minimum wave drag for a fixed volume. The classical theory of optimum shapes often does not produce geometries which can be used directly in actual flight vehicles. However, they do serve as a standard of the best which could be achieved under ideal circumstances--a sort of Carnot efficiency of aerodynamics.

A somewhat different approach to seeking optimum aerodynamic shapes which is suited to the CFD era is the use of numerical optimization methods. Generally speaking, numerical optimization procedures are search algorithms which seek to minimize (or maximize) an objective function subject to the specified constraints by systematically varying the free constants (design variables) which parameterize the system. Numerical optimization methods have been applied to many aerospace fields including structures, trajectories, guidance and control, propulsion, and preliminary design[17]. A recent article by Vanderplaats[18] gives a historical development of optimization methods in structures. The first application of numerical optimization to aerodynamic problems is that of Hague, Rozendall and Woodward[19] using linear panel methods. However, the most recent applications stem from the paper of Hicks, Murman and Vanderplaats[20] using transonic potential flow methods for airfoils. The farthest extensions of this work are the three-dimensional studies given in R-21-23. Also about the time the work of Hicks et al appeared, Parsons, Goodson and Goldschmied[24] applied the method to laminar flow control for underwater bodies.

Our concern in this article is to examine the application of numerical optimization to aerodynamic design. The attractiveness of this approach lies in its general applicability to practical "real world"

configurations. The aerodynamic analysis can be as complicated as a reliable CFD code can handle. The constraints may include not only geometric restrictions, but flow properties and off design performance. On the other hand limitations arise from computational efficiency and the ability to describe the body by a few parameters. It is our contention in this article that Aerodynamic Design by Numerical Optimization (ADNO) is potentially the most powerful design method using CFD. We will describe the method in more detail and cite illustrative examples. However, it is clear that the method is not yet practical for routine applications and we will conclude with suggestions for key areas requiring research.

In order to illustrate the numerical optimization technique, let us consider a hypothetical isolated wing design problem. First we define an objective function OBJ which is a single number representing the quantity we want to minimize. An example of an OBJ is the sum of the wave, friction, and induced drag. The definition of the objective function is up to the designer, but it must be expressible as a single number. If the designer wants to maximize some objective function, then OBJ is selected as the negative of that function. A frequently used objective function has been to minimize the deviation from a specified surface pressure distribution on the airfoil or wing. In this way an inverse problem is recast as an optimization problem.

Second we must define design variables which are free constants to be selected in order to minimize the objective function. The usual choices are coefficients of some functions describing the body shape. For example, if we start with a baseline wing, perturbations on different shapes could be added to the thickness and camber distributions at selected stations. The amplitudes of these perturbation shape functions are the design variables.

The wing twist and taper can also be design variables. In principle as many design variables (M) as desired may be used, but in practice the computational time increases approximately as M^2. Thus the selection of design variables becomes the challenge of finding as few free constants which describe as broad a class of body shapes as possible.

The M-dimensional space of design variables is called the design space. At each point in design space OBJ has a value. One can visualize this space with a topographical analogy, the contours of constant OBJ being elevation lines. The space is filled with hills and valleys, ridges and passes, and maybe even cliffs and cornices. The goal is to find the lowest point in design space. For a complicated problem, one can expect many local or relative minima. Cliffs correspond to discontinuities and cornices to non-uniqueness, both of which are bound to give mathematical difficulties. There is, of course, no guarantee that the flow field solution is unique. Recently, non-uniqueness has been demonstrated for transonic potential flow past an airfoil[25]. Let us consider the two-dimensional design space shown schematically in F-5. The design variables are a representative thickness ratio and taper ratio, and the objective function is some arbitrary performance parameter.

All designs have constraints which must be enforced. For our illustration, consider geometric constraints, aerodynamic constraints, and off-design constraints. These may be represented as barriers in design space as illustrated in F-6. Geometric constraints dictated by structures might correspond to minimum and maximum values of taper ratio and a minimum thickness. These are horizontal and vertical lines in design space or linear constraints. An aerodynamic constraint might be that the maximum adverse pressure gradient parameter on the wing be below a specified ceiling to

avoid separation and buffeting. Off design constraints might require no

leading edge separation at take-off and that the wing root bending moment in

a maneuver is limited. These latter constraints are shown as curved barriers

in design space as they most likely will be nonlinear functions of the

design variables. For each selection of the design variables, CFD codes or

other techniques must be available to evaluate the constraints and the

objective function.

The constrained design space for our illustration is the region

inside the constraint boundaries of F-6 and is called the feasible region.

The optimization problem is to find the point of minimum objective function

MIN within the feasible region. Numerical optimization procedures are

automated search processes which start from some initial point and seek

the minimum. Since the topology of the design space is not known ahead

of time, this is a "searching in the dark" task. There are a number of

algorithms available[26,27] including random walk, non-gradient, and gradient

methods to aid in this task. A complete review is outside the scope of

this paper.

The most widely used optimization method for aerodynamic design has

been the CONMIN algorithm developed by Vanderplaats[28]. It is a gradient

type constrained minimization algorithm based upon the method of feasible

directions. Starting at an initial point in design space, gradients of

OBJ are calculated. If no constraints are being violated, a search is made

in the direction of steepest descent until a new minimum is found or a con-

straint encountered. If the starting point is near an active constraint,

the direction normal to the constraint surface must be found so that the

search can be made in the direction which minimizes OBJ and stays in the

feasible region. At the conclusion of the search step, a new point in design space is established, new gradients are calculated, and another search performed, this time using a conjugate gradient method in place of steepest descent. The process continues until no reduction in OBJ is found. A more detailed description is given in R-29.

There are several general comments which can be made regarding the numerical optimization approach. First, the calculations can be lengthy with many searches and concomitant objective and constraint evaluations. Usually only a few constraints are "active" and that reduces the burden somewhat. Nevertheless, computational efficiencies of the algorithms are of critical importance. Second, the results are no more reliable than the aerodynamic models. Thus, design by optimization is no good unless the CFD codes have been thoroughly validated. Third, there is no guarantee a global minimum will be found. Although theoretically this may be a perplexing problem, it is of little practical importance. Each result is usually an improvement, and different starting points may be tried to seek new minimums. Finally, the method is very general. With the exception of the present man-in-loop procedures, we are not aware of any other method which can incorporate off-design constraints.

A number of publications have reported numerical optimization studies for low speed, multi-element, transonic and circulation controlled airfoils, transonic wings, a propeller, and a low drag underwater body. We will cite only representative results to illustrate various features of the method. In the majority of the studies, the optimization and CFD analysis codes were coupled as illustrated in F-7. The optimization code is the "driver." It passes values of the design variables to the CFD analysis code. In turn the CFD code returns values of the objective function and aerodynamic constraints.

If gradients are required, they are calculated from one-sided finite

difference formulas formed by individually perturbing each design variable

about the center point. The optimizer then evaluates gradients and search

directions, giving new values of design variables to the CFD code, etc.

For M design variables, (M + 1) CFD calculations are needed to compute

gradients. Order M searches are required, each taking about three CFD cal-

culations. Thus, order $3M^2$ CFD evaluations are needed. Usually an iterative

method is used in the CFD calculation to solve the flow equations. Since most

of the aerodynamic evaluations represent small changes from a previous solution,

convergence is much faster than for a regular analysis calculation[20]. Thus,

the total computational time is much less than $3M^2$ regular solutions.

Hicks, Murman, and Vanderplaats[20] considered the problem of minimizing

shockwave drag at transonic speeds. The CFD code was based on inviscid

transonic small disturbance theory for symmetric non-lifting bodies. Although

this is not a practical design problem, some interesting features were illus-

trated. F-8 shows a result where a virtually shock-free airfoil was obtained

starting from an arbitrary airfoil with a sizeable shockwave. Off design

drag results indicate a rapid increase past the design point. When an

additional constraint dictating off-design behavior was enforced (drag rise

between $M = .8$ and $M = .81$ less than 0.0015), the optimum airfoil has a weak

shock (F-9). F-10 illustrates an example for a supersonic flow with a bow

shockwave. In this case, the minimum drag body has a detached shockwave due

to the blunt nose. It is interesting that this body has a slightly lower

drag than a parabolic arc of revolution with the same enclosed volume, which

linear theory predicts should be the minimum drag body[16].

As a second example, we describe a study involving minimizing viscous

drag[24]. The application was to axisymmetric low drag under-water bodies

enclosing a specified volume. Drag minimization was obtained by delaying transition and avoiding separation. In this case the CFD model was a coupled potential flow panel method, boundary layer analysis, and standard transition prediction procedure. A polynomial description of the body with nine design variables was used and various geometric constraints were enforced. The constrained minimization procedure was of the random search variety. Results of the design indicated as "Body X-35" are shown in F-11 where they are compared to well known low drag body called the "Dolphin." The optimization procedure produced a body with a drag coefficient approximately 30% lower than the Dolphin for a Reynolds number of 10^7 (F-11a). Post-design calculations indicate this favorable effect is realized over a wide Reynolds number range (F-11b).

The basic procedure of the Hicks, Murman, Vanderplaats study was extended to more practical inviscid transonic flow calculations[29] by using a CFD program based on the transonic full potential equation for lifting airfoils. Researchers at Lockheed-Georgia Co.[30] applied the method to a redesign study of the C141 airfoil. Two redesigns were done. The first involved modifying only the first 12% of the upper surface. The second involved modifying the entire upper surface. In both studies, the objective was to reduce the drag at the cruise Mach number of 0.74. For the second modification the design was constrained to have the same maximum thickness and nose bluntness as the baseline C141 airfoil, approximately the same lift coefficient, and also an off-design constraint that the wave drag at $M = .72$ be less than 0.0020. This was to avoid "drag-creep" which can result from doing a single point design.

For the first study where the leading edge region was modified, viscous effects are negligible and the design methodology was borne out by

wind tunnel verification (F-12). The researchers state: "The wind tunnel test data showed that a 7% improvement in (ML/C) may have resulted from the modification of the upper surface of the forward 12% of the airfoil. In addition to producing an efficient airfoil modification, numerical optimization required about half the computational time and resulted in a 25% reduction in engineering hours when compared to a conventional trial-and-correction approach." For the second study involving modification of the entire upper surface, viscous effects are important. An attempt was made to correct for them but the result was unsuccessful. The wind tunnel tests showed a higher drag than the base line airfoil (F-12). The negative result of the second study was due to the inability of the CFD code to model the viscous effects. In other studies (e.g. R-22) where a simple displacement thickness was added, the airfoil design process was in better agreement with experimental data.

Several three-dimensional numerical optimization wing designs have been reported[21-23]. The study by Haney, Johnson, and Hicks[22] sought to redesign the A-7 wing to produce substantial improvements. The CFD code used was an inviscid transonic full potential equation method (FLO22) with a simple two-dimensional viscous displacement surface added. OBJ for this case was the deviation from a specified surface pressure distribution; that is, an inverse problem was solved. A total of 120 design variables were used. At each of five span stations, the twist, trailing edge camber, and twenty-two surface shape functions were used as design variables. It was prohibitive to vary all of these design variables simultaneously and instead, each section was optimized sequentially starting at the root and moving to the tip. The design goal of the first study (Wing No. 1) was to increase the thickness by 71% with the same drag divergence Mach number. The design

goal of the second study (Wing No. 2) was to increase the thickness by 28%, reduce the induced drag by 25%, and maintain the same drag divergence Mach number. These design goals together with two-dimensional airfoil tests led to the specification of the desired wing pressure distribution which formed the objective function. The planform shapes were preset by preliminary design studies. F-13 shows the planform geometries and results of the wind tunnel tests. Both wings show substantial improvement in drag characteristics compared to the original A-7 wing without any degradation in drag divergence Mach number. The results of the numerical optimization procedure were successful. Although one of the three-dimensional inverse methods[7-11] may have been more computationally efficient, the application of numerical optimization was relatively straightforward for this case.

As a final example, we cite the results of a U.S. Air Force sponsored study[23] with Lockheed-Georgia Co. and Grumman Aerospace Corp. The purpose was "to develop and validate a new transonic wing design procedure using numerical optimization techniques." Both fighter and transport configurations were considered in a three-phase study involving design, test, and evaluation. Although many limitations were encountered, the concluding remarks from the transport study include: "We have developed a new transonic wing design method using the numerical optimization scheme. We have also shown that new computational methods offer a means for the aerodynamic design of wings with transonic performance superior to that which could be obtained using previous design techniques. The method is relatively easy to use, and it is compatible with established industry design procedures. By using the

new method, a 40% to 50% reduction in the cost associated with wing cruise aerodynamics design is obtainable." The latter statement was made despite an estimated 5-10 hour computation time on a CDC-7600 computer. The fighter configuration study resulted in the statement: "The results described here show both the benefits and hazards of numerical optimization for aerodynamic design. The numerical optimization will work best when both the flowfield analysis and numerical model of the design problem are accurate. Although greater computer resources are generally needed for more complex and accurate analysis, the cost of numerical optimization would still be less than that of additional wind tunnel testing. As both computer and analysis code capability increase, numerical optimization will take a greater role in aerodynamic design."

The above examples illustrate many advantageous features of aerodynamic design by numerical optimization (ADNO) and call attention to some of the limitations. Clearly the basic CFD model must produce reliable results before optimization can be used. However, with the enormous effort being expended to develop such codes, it can be expected that the future is bright in this category. The efficiency of the procedure depends in part on using aerodynamic "smartness" to specify the objective function, constraint function, and design variables. It is desirable to pick functions which are aerodynamically "well conditioned." Perhaps some sort of adaptive parameterization can be developed to select those design variables which are making the biggest contribution and "turn off" the unimportant ones. Another possibility might be to further exploit the power of inverse methods by combining them with optimization techniques as suggested by Garabedian[31].

The most important limitation at present appears to be the large computer resources which are required. However, we note the experience of the

Lockheed-Georgia[30] team that if a careful accounting is done of the total

engineer and computer hours to do a design, ADNO may not be as expensive

as it appears. With the yearly reductions in computer costs and increase

in computer capability, this situation should become less severe. Be

that as it may, it is desirable to develop algorithms which are more

efficient. The cited ADNO calculations have been performed by taking

existing CFD and optimization codes and combining them in "black-box"

fashion (F-7). By far, the most time is spent in the CFD code doing aero-

dynamic calculations. Two suggestions can be made for more efficient

approaches.

The first is to eliminate the requirement to execute the complete

CFD calculation for every aerodynamic evaluation and to replace it by a

suitable approximate solution (F-14). For gradient calculations and as

the minimum is approached, each new set of design variables should produce

only a small change in the aerodynamic solution. Thus the complete CFD

code need only be used for calculations producing large changes and an

approximate calculation may be done for small changes. The selection

criteria may be preset or done adaptively. Some demonstrations along this

line are reported by Vanderplaats[32], Stahara, Elliot, and Spreiter[33],

Bristow, Hawk, and Thomas[34], and Peeters[35].

The second suggestion is to combine the optimization and CFD

algorithms into a one-step procedure in which the only converged solution

is the final answer. One technique for solving large sets of discrete

equations is to minimize the residuals. By using Lagrange multipliers,

objective functions and constraints might be incorporated directly into

the residual minimization. Labrujere[36] reports results using this approach.

15

It is hoped that this article will stimulate interest and creative thinking about the enormous potential of optimization procedures for aerodynamic design and how it can be developed into a practical approach.

References

1. Graves, R.A.,Jr., "Computational Fluid Dynamics: The Coming Revolution." Astronautics and Aeronautics, March 1982, pp 20-28.

2. Bauer, F., Garabedian, P., Korn, D. and Jameson, A., Supercritical Wing Sections-II, Springer, New York, 1975.

3. Miranda, L.R., "A Perspective of Computational Aerodynamics from the Viewpoint of Airplane Design Applications." AIAA paper 82-0018, Jan. 1982.

4. Boppe, C.W. and Aidala, P.V., "Complex Configuration Analysis at Transonic Speeds." AGARD CP-285, May 1980, paper 26.

5. Yu, N.J., "Grid Generation and Transonic Flow Calculations for Three Dimensional Configurations." AIAA paper 80-1391, July 1980.

6. Thompkins, W.T.,Jr., "A FORTRAN Program for Calculating Three-Dimensional, Inviscid, Rotational Flows with Shock Waves in Axial Compressor Blade Rows." MIT GT&PDL Report No. 162, Sept. 1981.

7. Henne, P.A., "An Inverse Transonic Wing Design Method." AIAA paper 80-0330, Jan. 1980.

8. Davis, W.H.,Jr., "Technique for Developing Design Tools From the Analysis Methods of Computational Aerodynamics." AIAA J, 13:9, pp 1080-1087.

9. Shankar, V., "A Full Potential Inverse Method Based on a Density Linearization Scheme for Wing Design." AIAA paper 81-1234, June 1981.

10. Bristow, D.R. and Hawk, J.D., "A Mixed Analysis and Design Surface Panel Method for Two-Dimensional Flow." AIAA paper 82-0022, Jan. 1982.

11. Garabedian, P. and McFadden, G., "Design of Supercritical Swept Wings," AIAA J, 20:3, March 1982, pp 289-291.

12. Melnik, R. and Volpe, G., "The Role of Constraints in the Inverse Design Problem for Transonic Airfoils." AIAA paper 81-1233, June 1981.

13. Nieuwland, G.Y. and Spee, B.M., "Transonic Airfoils: Recent Developments in Theory, Experiment, and Design." In Annual Reviews of Fluid Mechanics, Vol 5, 1973 Annual Reviews, Inc., Palo Alto, CA.

14. Chin, W.C. and Rizzetta, D.P., "Airfoil Design in Subcritical and Supercritical Flows." J.Appl.Mech. Vol 46, No. 4, Dec. 1970, pp 761-766.

15. Fung, K.Y., Seebass, A.R., Dickson, L.J. and Pearson, C.F., "An Effective Algorithm for Shock-Free Wing Design." AIAA paper 81-1236, June 1981.

16. Miele, A., Theory of Optimum Aerodynamic Shapes, Academic Press, New York, 1965.

17. Ashley, H., "On Making Things Best--Aeronautical Uses of Optimization," AIAA paper 81-1738, Aug. 1981.

18. Vanderplaats, G.H., "Structural Optimization--Past, Present and Future," AIAA J, Vol 20, No. 7, July 1982, pp 992-1000.

19. Hague, D.S., Rozendaal, H.L. and Woodward, F.A., "Application of Multivariable Search Techniques to Optimal Aerodynamic Shaping Problems." J.Astro.Sci., Vol XV, No. 6, Nov-Dec 1968, pp 283-296.

20. Hicks, R.M., Murman, E.M., and Vanderplaats, G.N., "An Assessment of Airfoil Design by Numerical Optimization," NASA TM X-3092, July 1974.

21. Hicks, R.M. and Henne, P., "Wing Design by Numerical Optimization," AIAA paper No. 77-1247, 1977.

22. Haney, H.P., Johnson, R.R. and Hicks, R.M., "Computational Optimization and Wind Tunnel Test of Transonic Wing Analysis," AIAA paper 79-0080, Jan. 1979.

23. Srokowski, A.J., Lores, M.E., and Aidala, P., "Numerical Aircraft Design Using 3-D Transonic Analysis with Optimization," AFWAL-TR-81-3091, Vols I-III, Aug. 1981.

24. Parsons, J.S., Goodson, R.E., and Goldschmied, F.R., "Shaping of Axisymmetric Bodies for Minimum Drag in Incompressible Flow." J.Hydronautics, Vol 8, No. 6, 1974, pp 100-107.

25. Steinhoff, J. and Jameson, A., "Multiple Solutions of the Transonic Potential Flow Equation," AIAA paper 81-1019, June 1981.

26. Bekey, G.A., "System Identification--An Introduction and a Survey," Simulation, October 1970, pp 151-166.

27. Fox, R.L., Optimization Methods for Engineering Design. Addison-Wesley, Reading, MA, 1971.

28. Vanderplaats, G.N., "CONMIN--A FORTRAN Program for Constrained Function Minimization," NASA TMX-62,282, Aug. 1973.

29. Vanderplaats, G.N., Hicks, R.M., and Murman, E.M., "Application of Numerical Optimization Techniques to Airfoil Design," NASA SP-347, March 1975, pp 749-763.

30. Lores, M.E. Burdges, K.P., and Shrewsbury, G.D., "Analysis of a Theoretically Optimized Transonic Airfoil," NASA CR 3065, Nov. 1978.

31. Garabedian, P., "Computational Fluid Dynamics of Airfoils and Wings," presented at Symp. on Transonic, Shock and Multidimensional Flows, U. of Wisconsin, May 1981.

32. Vanderplaats, G.N., "An Efficient Algorithm for Numerical Airfoil Optimization," AIAA J.Aircraft, Vol 16, No. 12, Dec. 1979, pp 842-847.

33. Stahara, S., Elliott, J.P., and Spreiter, J.R., "Development of a Multiple-Parameter Nonlinear Perturbation Procedure for Transonic Turbomachinery Flows: Preliminary Application to Design/Optimization Problems," Nielsen Eng. and Rsch Inc. TR 276, May 1982.

34. Bristow, D.R., Hawk, J.R., and Thomas, J.C., "Subsonic 3-D Surface Panel Method for Rapid Analysis of Multiple Geometry Perturbations," AIAA paper 82-0993, June 1982.

35. Peeters, M.F., "Computational Methods for Aerodynamic Design Using Numerical Optimization," SM Thesis, MIT Dept. of Aero and Astro, Feb. 1983.

36. Labrujere, Th.E., "Airfoil Design by the Method of Singularities via Parametric Optimization of a Geometrically Constrained Least Squares Object Function," NLR TR 76139, Feb. 1976.
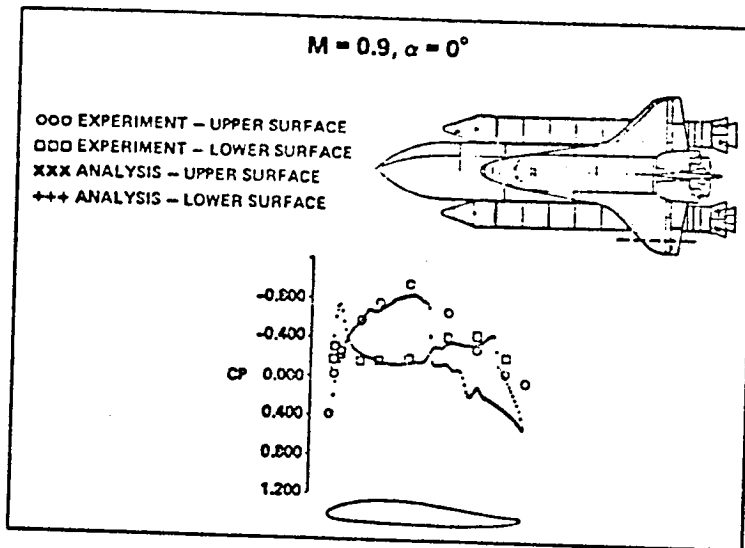
F-1. Comparison of computed and measured pressure
distribution on shuttle orbiter wing in
launch configuration (R-4)

INBOARD STATION

C_p

-.5

0.0

● TEST DATA

—— WING ALONE CODE

- - - WING-BODY CODE

.5

C_p

-1.0

OUTBOARD STATION

-.5

0.0

.5

0.0   .5   1.0

X/C

y

x
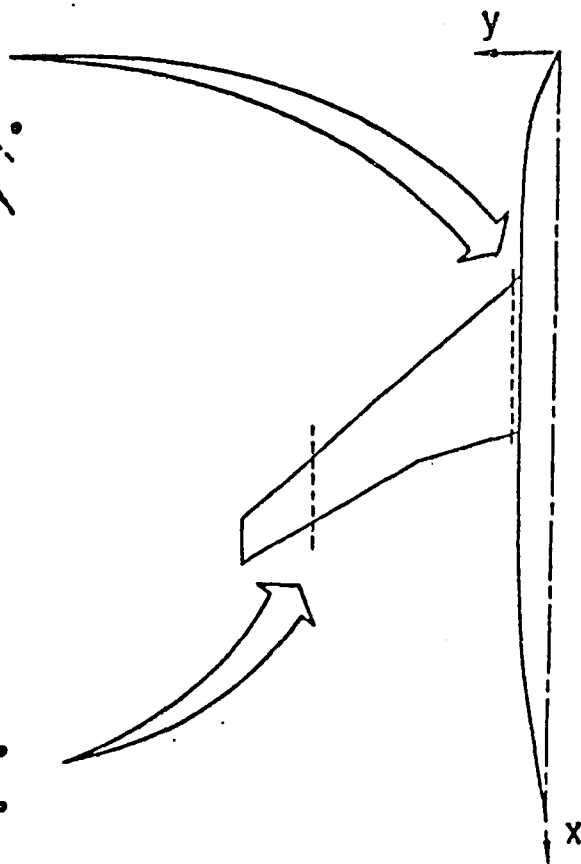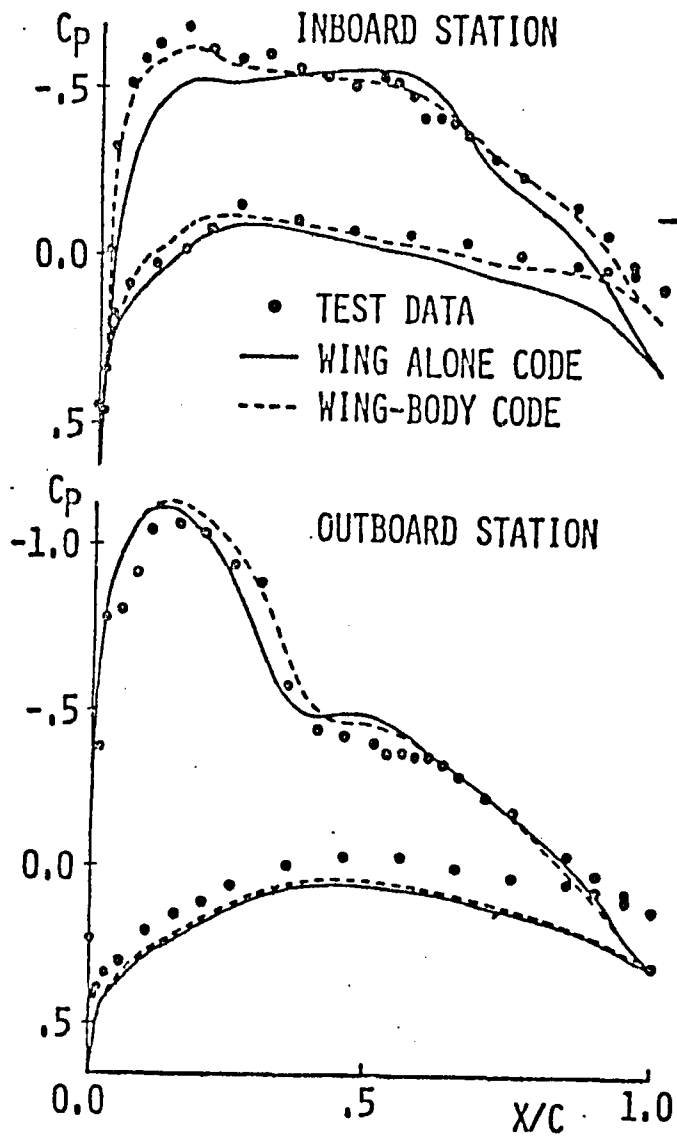
F-2. Calculation showing the effect of
fuselage on a transonic transport
wing pressure distribution (R-5)

Weak
Oblique
Shock

Normal
Shock

Sonic Line

Strong
Oblique Shock.
Maximum
Turning Angle
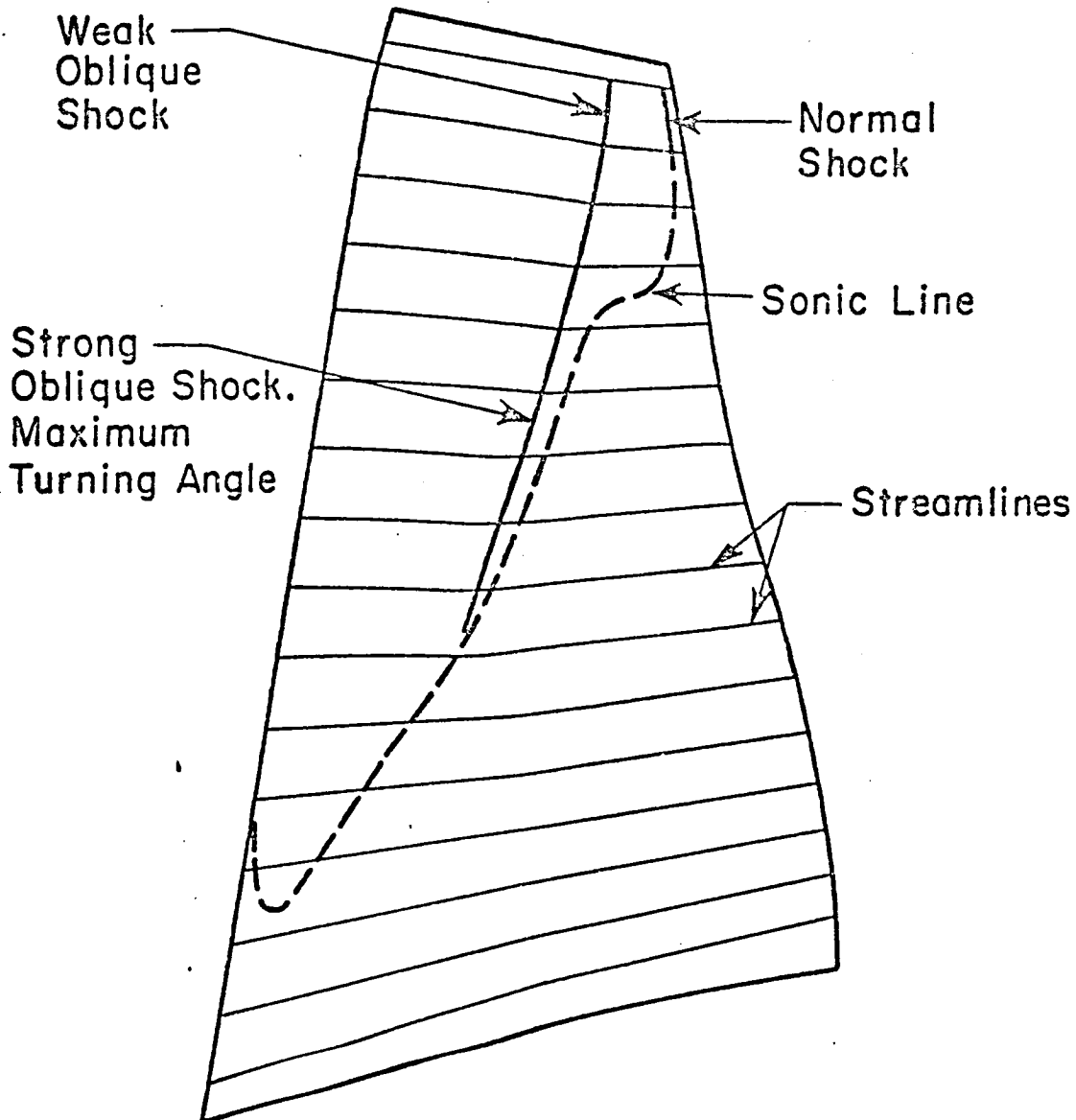
Streamlines

F-3. Suction surface shock wave, streamline, and sonic line
      structure from three-dimensional solution of Euler
      equations for NASA low aspect ratio rotor (R-6)

EXPERIMENTAL/COMPUTATIONAL DESIGN
EVOLUTION

EXPERIMENTAL
AERODYNAMIC DESIGN
G-II (1966)

24 WINGS TESTED

COMPUTATIONAL
AERODYNAMIC DESIGN
G-III (1979)

1 WING/WINGLET TESTED
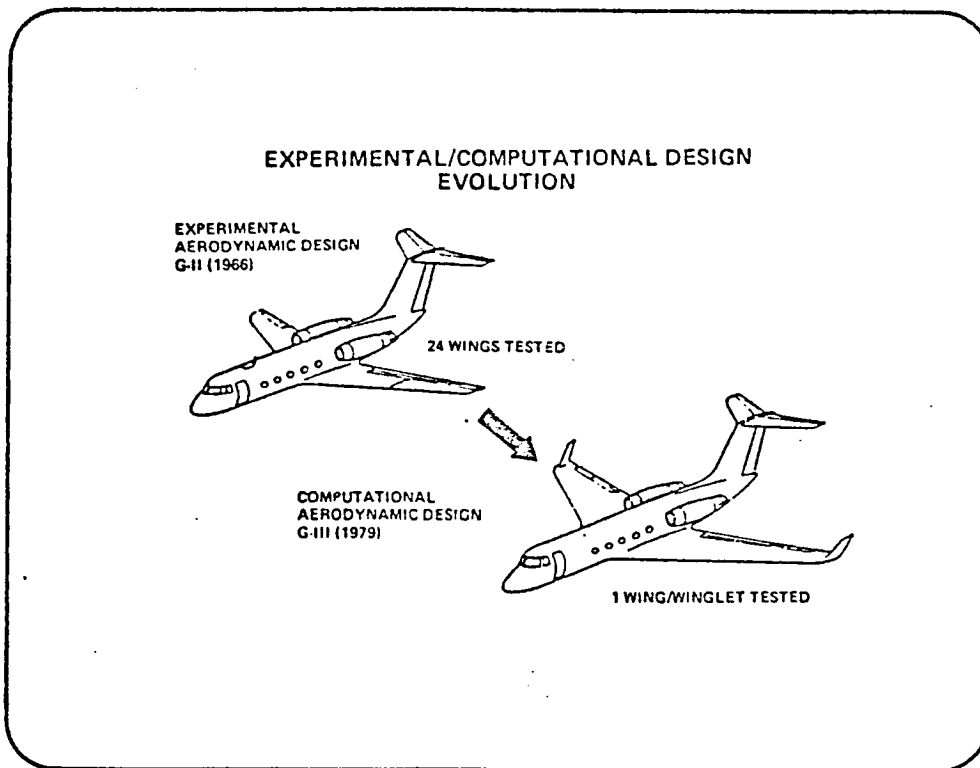
F-4. Impact of CFD on design of Grumman Gulfstream III wing

THICKNESS RATIO
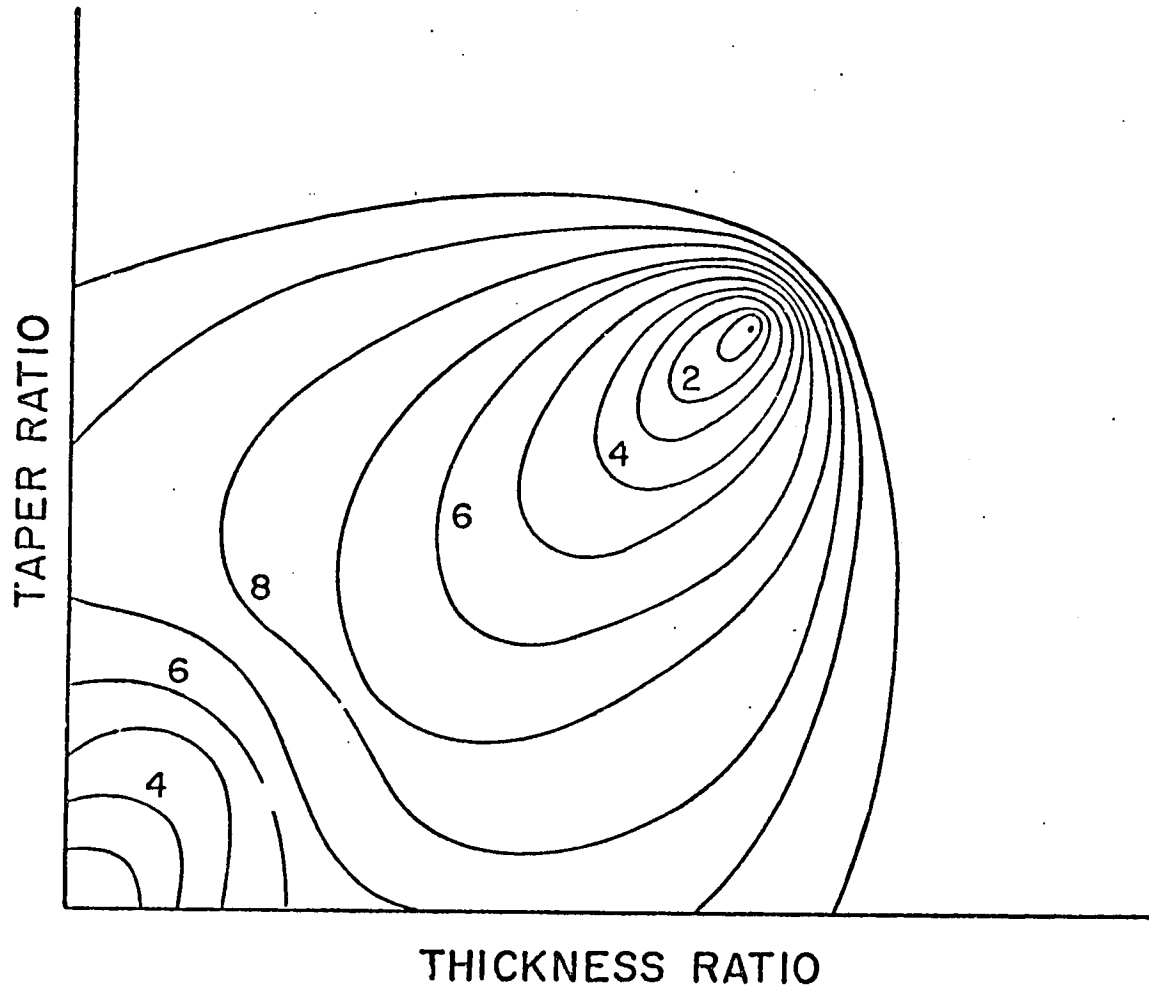
F-5. Unconstrained design space. Contours represent curves of constant objective function.
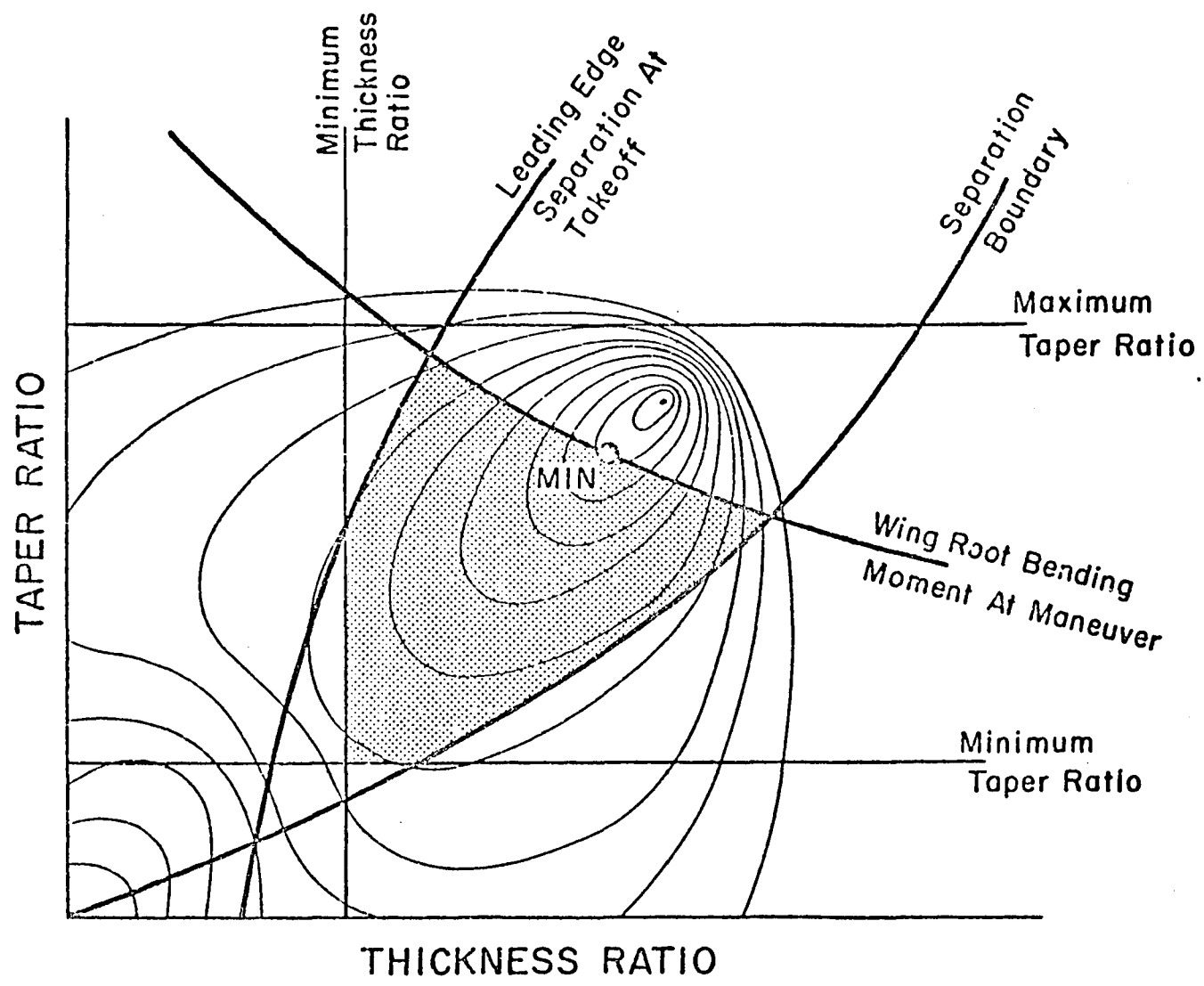
F-6. Constrained design space. The feasible region is the shaded area enclosed by all the constraint boundaries. MIN is point of optimum design.
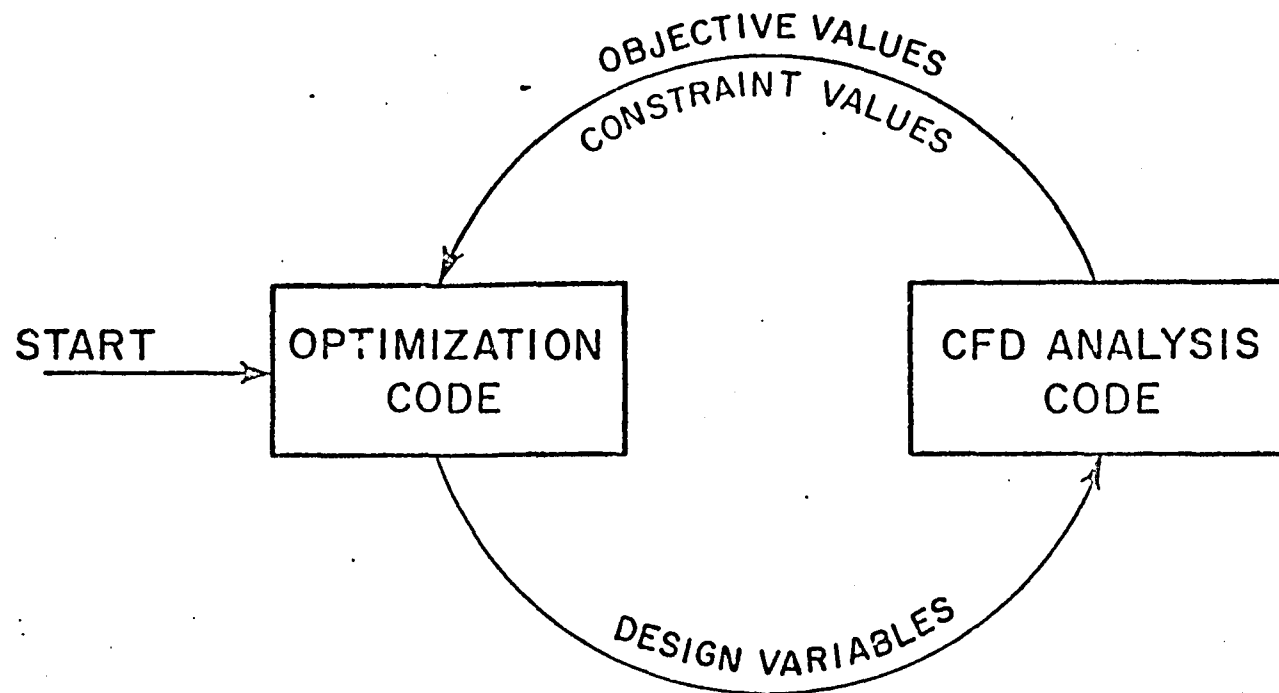
F-7. Standard coupling of optimization and CFD analysis codes

(a)

C_p

C_p^*

t/c, percent chord

——— INITIAL AIRFOIL (ARBITRARY), $C_D = 0.0083$, $V = 0.721$

– – – FINAL AIRFOIL, $C_D = 0.0002$, $V = 0.779$

x/c, percent chord

(b)

C_D

M_{DESIGN}

M_{CRIT}

M
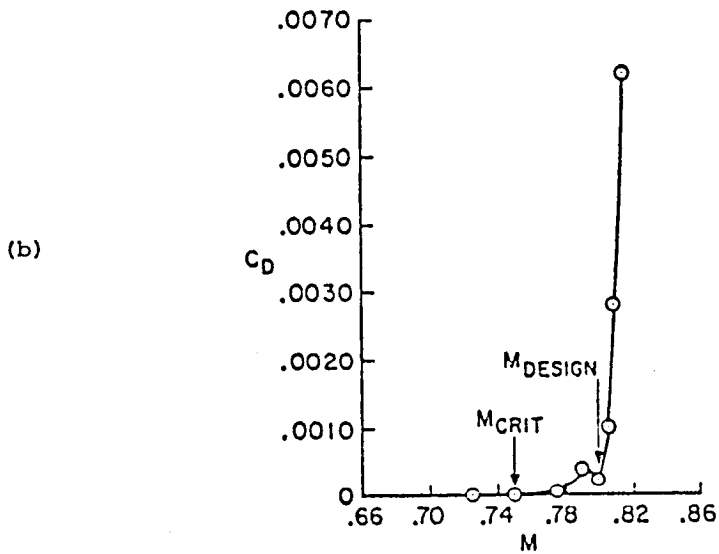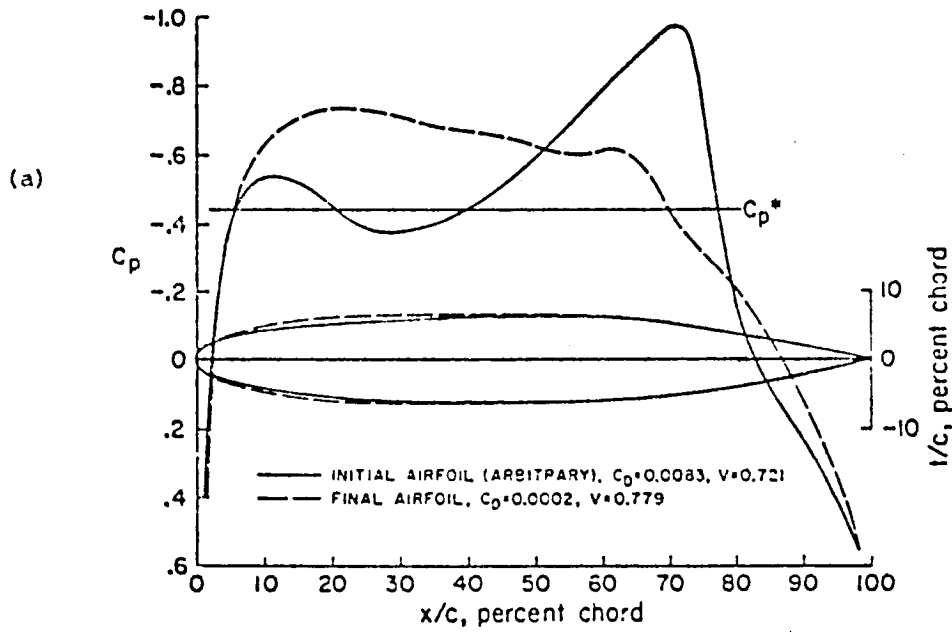
F-8. Wave drag minimization for transonic airfoil with constraints
on minimum thickness and enclosed volume. (a) Initial and
final airfoil shapes and pressures. (b) Drag rise character-
istics of designed airfoil (R-20)

(a)

C_p

STARTING AIRFOIL, $C_D = 0.0002$, $V = 0.779$
FINAL AIRFOIL, $C_D = 0.0002$, $V = 0.748$

C_p^*

t/c, percent chord

x/c, percent chord

(b)

C_D

AIRFOIL
△ MODIFIED
○ ORIGINAL

M
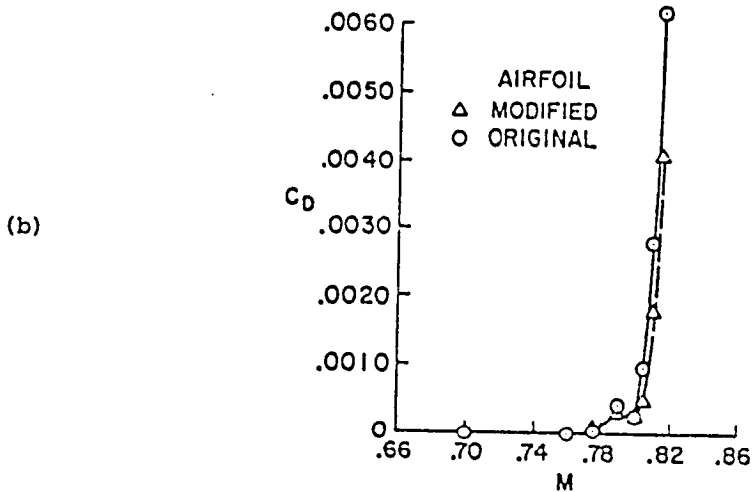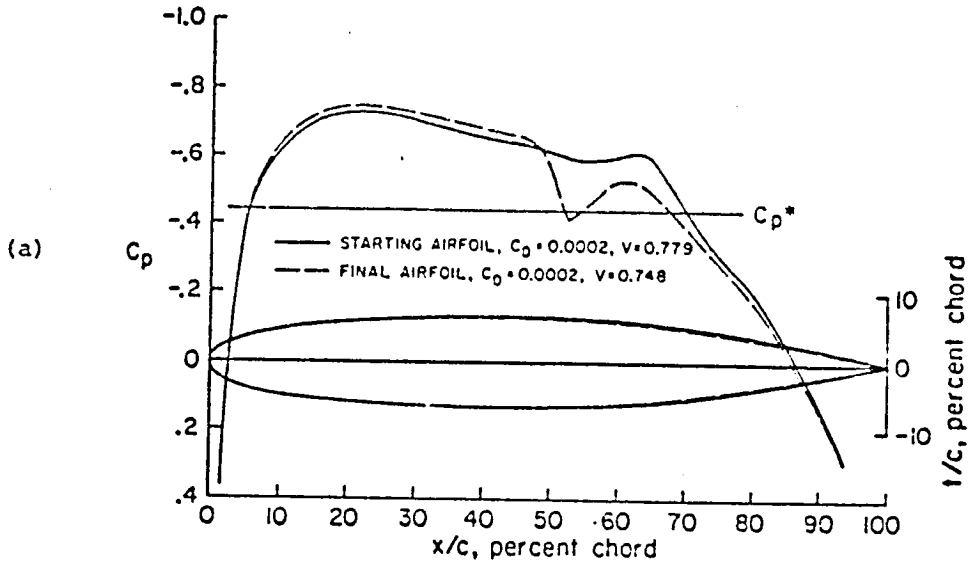
F-9. Effect of off design drag rise constraint on airfoil design
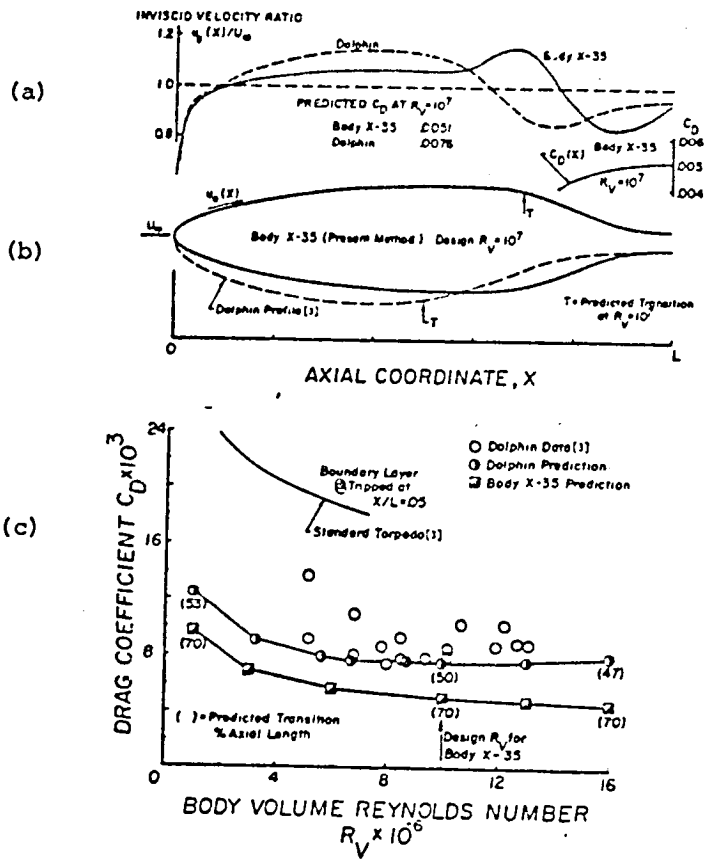of F-8. Constraint is $C_{D_{M=.81}} = - C_{D_{M=.80}} \leq 0.0015$.

(a) Original and modified airfoil shapes and pressures.
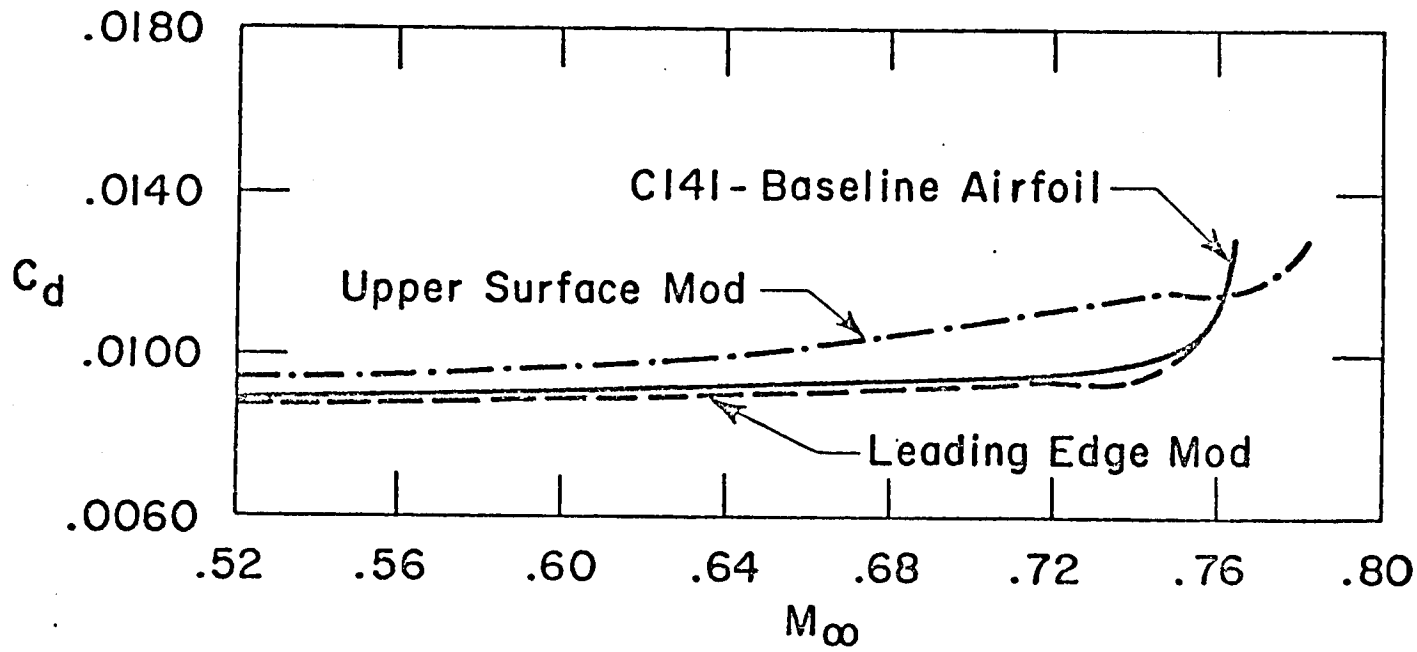(b) Drag rise characteristics (R-20)
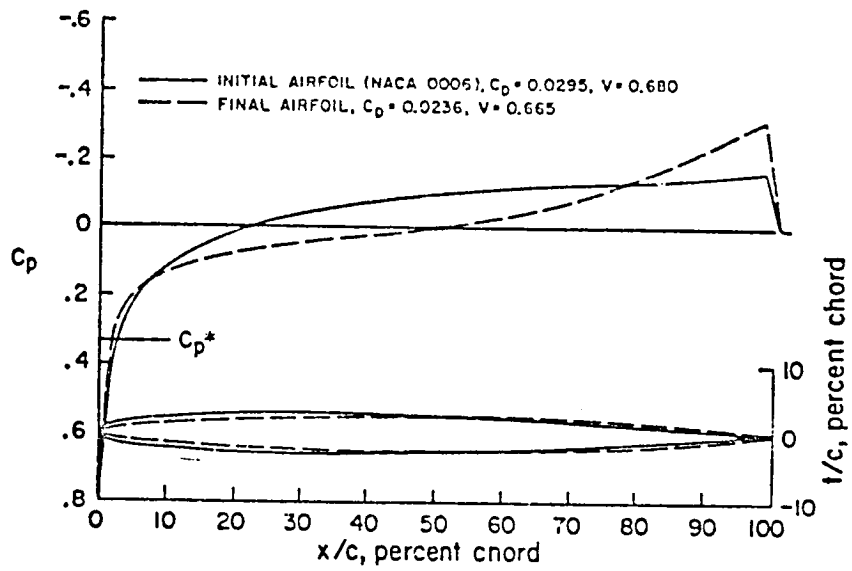
F-10. Wave drag minimization for supersonic flow past
non-lifting airfoil with detached bow shock wave.
Constraint on enclosed volume. $M_\infty = 1.3$ (R-20)

F-11. Comparison of Dolphin body with low drag laminar flow
axisymmetric body designed by numerical optimization.
(a) Surface velocity, (b) Body shape, (c) Drag
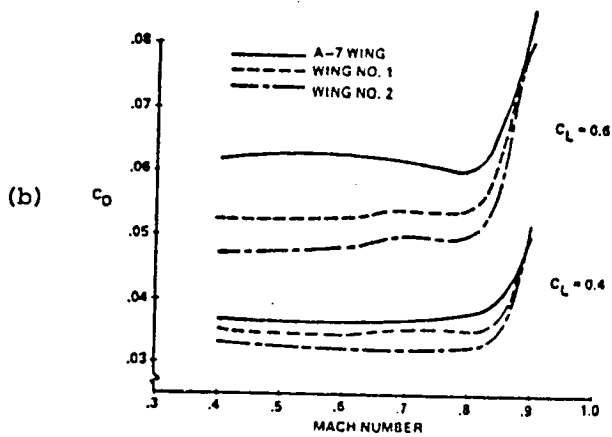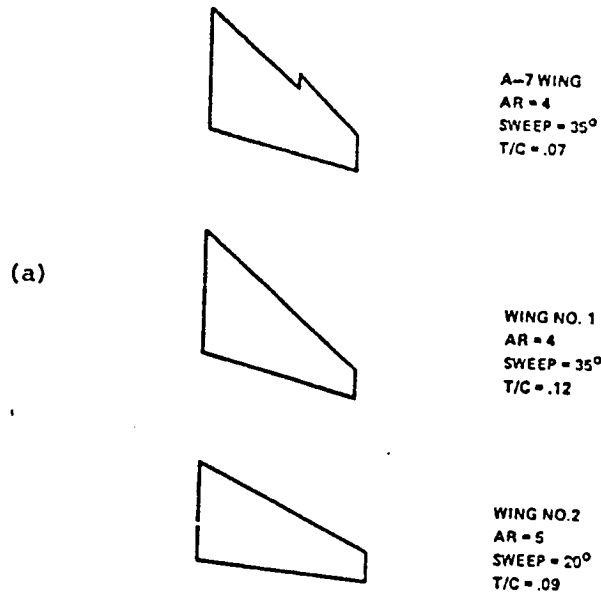coefficient vs. Reynolds number (R-24)

F-12. Measured airfoil performance for baseline C-141 airfoil compared with two modifications designed by numerical optimization (R-30)

(a)

A—7 WING
AR = 4
SWEEP = 35°
T/C = .07

WING NO. 1
AR = 4
SWEEP = 35°
T/C = .12

WING NO.2
AR = 5
SWEEP = 20°
T/C = .09

(b)

C_D

A—7 WING
WING NO. 1
WING NO. 2

C_L = 0.6

C_L = 0.4

MACH NUMBER

F-13. A-7 wing and two optimized modified wing designs.
(a) Planform shapes, (b) Experimental drag rise
at two lift coefficients (R-22)
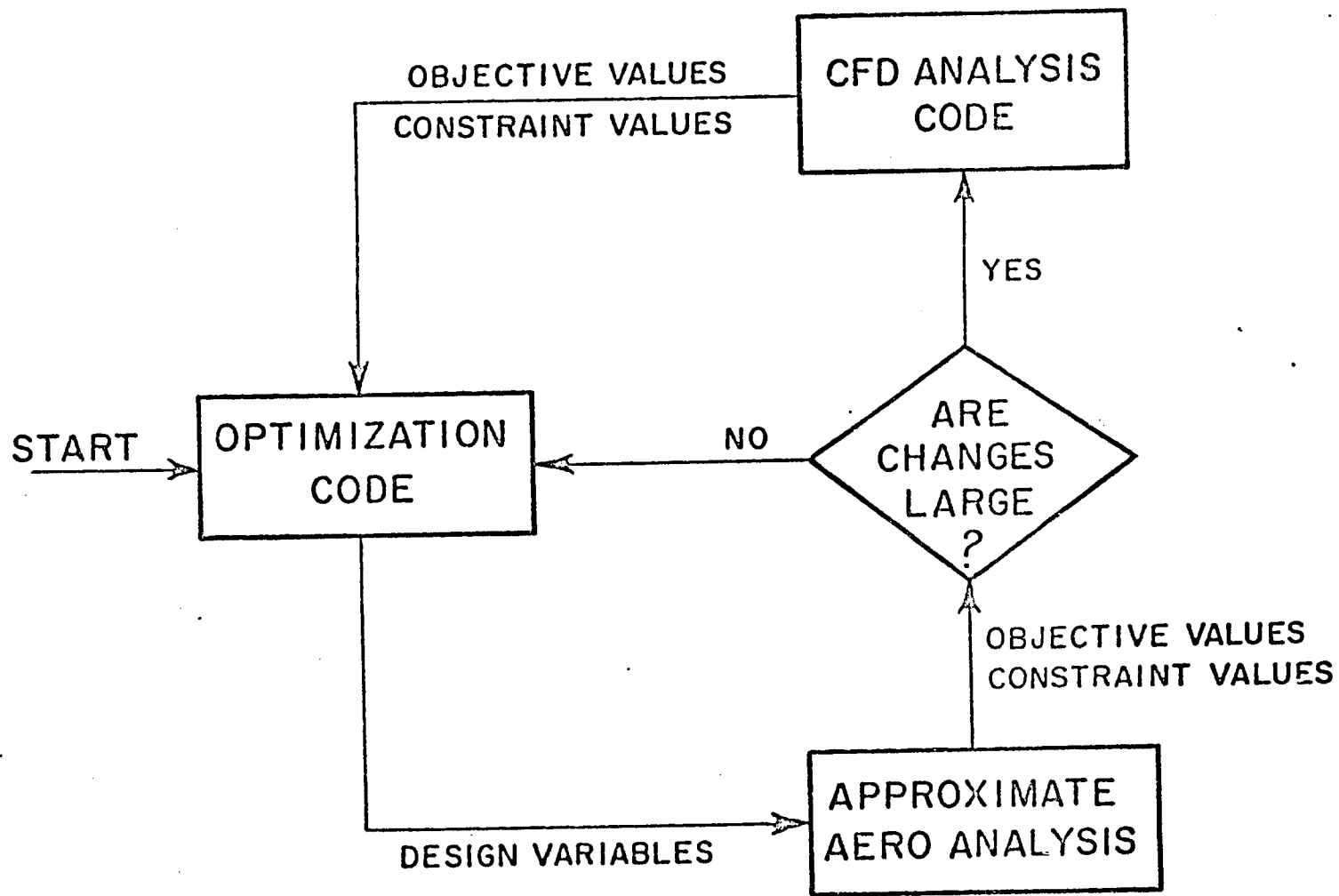
F-14. Possible coupling of optimization code with aerodynamic analysis methods
which could reduce computer time

END

DATE

FILMED

SEP 5 1984